International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 9, Issue 3, Jun 2019, 631-636

A COMPARATIVE ANALYSIS OF THE FLOW PATTERN USING FLUE GASES AND WATER INSIDE ECONOMIZER

A. ADAIKALARAJ[1], M. SUNDARARAJ[2] & N. KALAIMANI[1]

[1]*Assistant Professor, Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Chennai, Tamil Nadu, India*

[2]*Head and Associate Professor, Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Chennai, Tamil Nadu, India*

ABSTRACT

This project is an effort to analyze the velocity distribution inside the economizer tube by changing the working fluids. Flue gases and water were considered as the working fluid in our study. The velocity plays a key role in heat transfer by convection method. This paper presents the computational analysis of economizer normally employed in thermal power plant sectors. The reason behind the study of velocity distribution is, the flue gases exhibits greater velocity at lower bend areas when compared with upper bend regions. This leads to erosion in upper bends while employing flue gas as the working fluid. The computational model was resolved using conventional CFD techniques. The three dimensional economizer coils was designed using CATIA V5 software and the temperature, pressure and the inlet velocity of the fluid flow were given as the boundary conditions for our analysis.

KEYWORDS: Flue Gases, Economizer, Velocity Distribution & Erosion

Received: Mar 14, 2019; **Accepted**: Apr 04, 2019; **Published:** May 03, 2019; **Paper Id.:** IJMPERDJUN201969

Original Article

INTRODUCTION

The heat transfer surfaces are the primary elements of a boiler which actually transfers the heat by means of the circulation of the working fluid. The modern heat transfer surfaces are economizers, air pre-heaters, super heaters, evaporators, furnaces and so on. To use more efficient energy systems, leads the increasing cost. This in turn leads to the growth in researches in the field of augmenting heat transfers. Several techniques including passive and active methods were developed for enhancing the heat transfer. Some of them are rifled tubes, central helically twisted ribbons and full helical ribbons and so on. These are found to be the most powerful heat transfer enhancing technology. This study is an effort to understand the heat transfer rate while changing the working fluids. The computational analysis by changing the working fluid was done by using ANSYS software packages and the geometry was designed using CATIA V5 software. Economizer actually performs a key function in transferring heat to the overall boiler. The economizer shows better thermal efficiency characteristics by recovering the energy by heating the boiler feed water. Normally, economizer is a tubular structure used to preheat the working fluid. Maximum allowable flue gas velocity is the key design criteria for economizer. Higher velocities of the flow of working fluid shows better heat transfer rate and reduces the initial investments also. Heat flow distribution is the major problem in the economizers employed in thermal plants. CFD analysis of economizer can be useful to gain the knowledge about the gas flow distribution. Efforts were made to obtain the velocity and temperature distribution of flue gases. Our work mainly concentrates on the bend regions. In our work the flow

patterns and flow parameters were observed and recorded.

MODELLING AND SIMULATIONS

The modelling tool used for designing the economizer tube is CATIA V5. CATIA is widely used to design and model the objects which could be further imported in the numerical tools to carry out simulations. The length of the coil is 8200mm, the pitch of the coil is 115mm, the diameter of the coil is 44.5mm, and the thickness of the coil is 4.5mm. The coil is having total number of turns as 9 and the turn diameter was chosen as28.75mm.

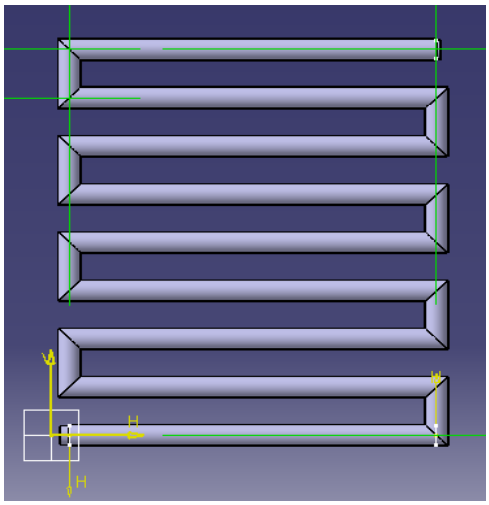


Figure 1: Economizer Model

The model was imported to ICEMCFD software for discretization. A several set of finite volumes were constructed based on the Gauss divergence theorem. This process actually converts the volume integrals, including the gradient operators and divergence to surface integrals. The accuracy of the simulation depends on the mesh quality. The discretized model is shown in the figure shown below.

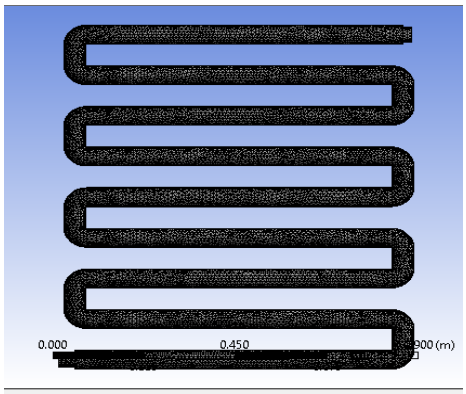
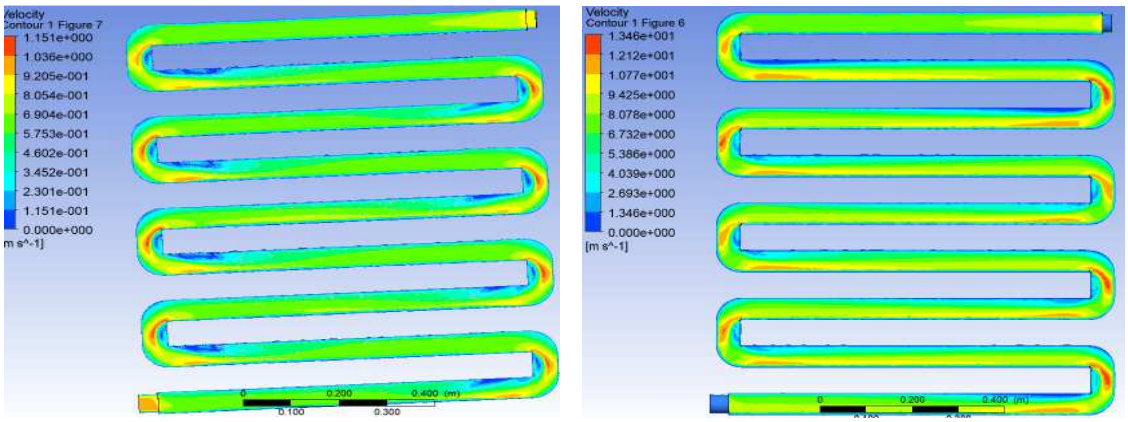


Figure 2: Discretized Model of Economizer

RESULTS AND DISCUSSIONS

ANSYS CFX tool is used to predict the pressure, velocity & temperature distribution inside the economizer tube. The analyses were done for two different cases including water as working fluid and flue gases as working fluid. Our work is concentrated to obtain the flow nature and the velocity distribution near the bend regions. The velocity distribution shows that the hot flue gas loses as it moves in a downward direction.

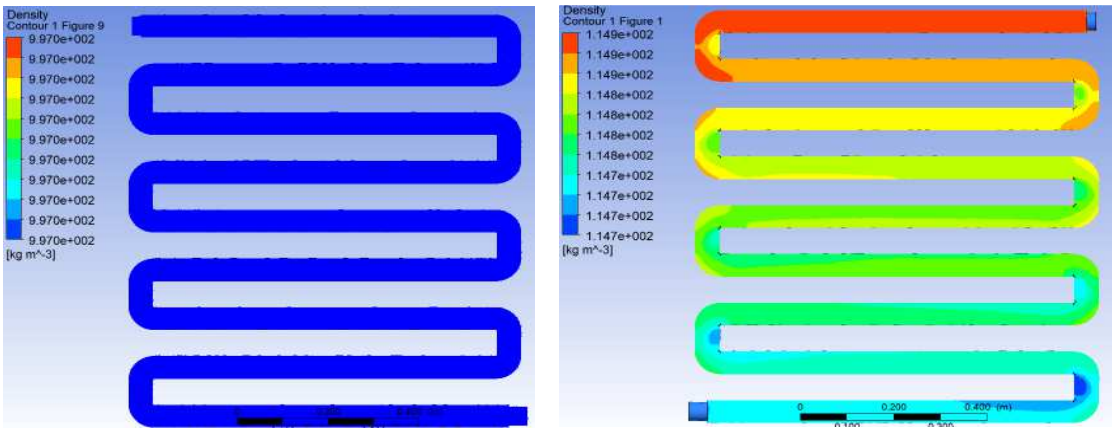
Velocity Contour



For Water **For Flue gases**

Figure 3: Velocity Contour for Economizer

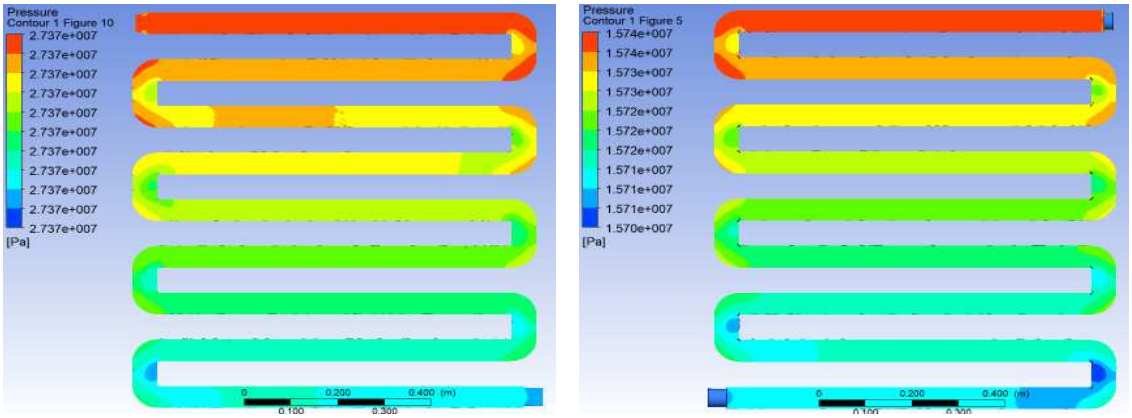
Density Contour



For Water **For Flue gases**

Figure 4: Density Contour for Economizer

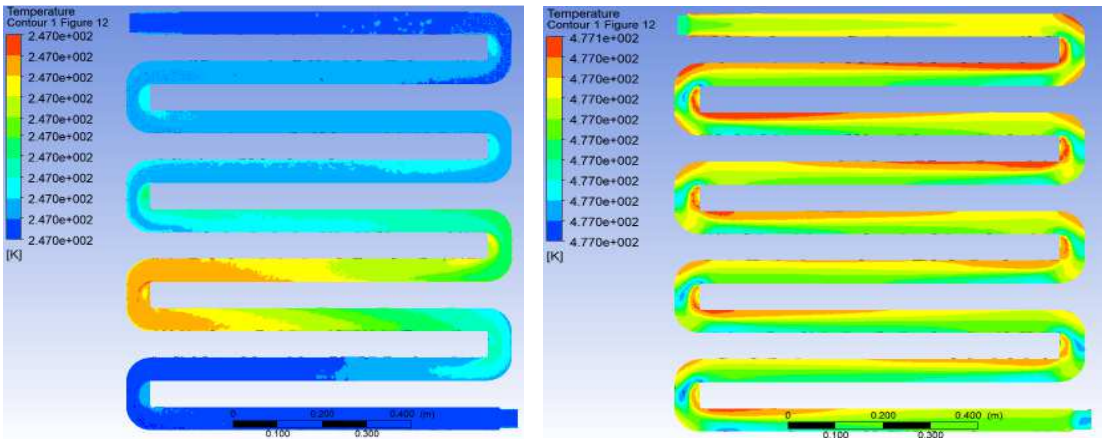
Pressure Contour



For Water **For Flue Gases**

Figure 5: Pressure Contour for Economizer

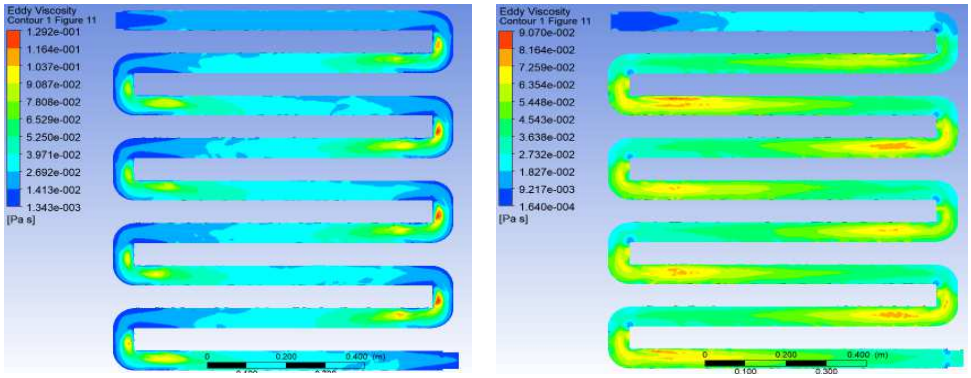
Temperature Contour



For Water **For Flue Gases**

Figure 6: Temperature Contour for Economizer

Eddy Viscosity Contour



For Water **For Flue gases**

Figure 7: Eddy Viscosity for Economizer

CONCLUSIONS

Economizer performs a key function in providing high overall boiler thermal efficiency by recovering the low level energy from the flue gas before it is exhausted to the atmosphere. In thermal plants, economizer recovers the energy by heating the boiler feed water. The flue gases exhibit greater velocity at lower bend areas when compared with upper bend regions.

REFERENCES

1. *D. Patil,P. R. Baviskar, "Optimization of EconomiserDesign for the Enhancement of Heat Transfer Coefficient"*
2. *J B Kitto and S C Stultz "Steam – its generation and use"The Babcock &Wilcox Company, 40th edition*
3. *Charles E Baukal "Combustion Handbook", the John ZinkCompany. [4] John D Anderson "Computational FluidDynamics the basics and application" McGraw-Hill,Dept.of Aerospace Engineering, University of Maryland.*
4. *Suhas V Patankar "Numerical Heat Transfer and FluidDynamics".*

5. *P.K.NAG"Power Plant Engineering", 2006, TataMcGraw-Hill Publication.*
6. *PMV Subbarao "CFD Analysis of Flue Gas Duct", 05 Jan2010, IIT, Delhi.*
7. *V Ganapathy "Industrial Boilers and Heat Recovery SteamGenerators, CRC", ABCO industries, Abilene, Texas,USA.*
8. *Huang, H. C., & Lee, Y. D. (2017). A Study of Indigenous Measuring Factors for Employer Brand Attractiveness in Taiwan: Comparative Analysis of Academy and Industry Experts. IMPACT: International Journal of Research in Business Management (IMPACT: IJRBM), 5(11), 11-20.*
9. *S. Jayanti "Techniques for flow passage optimization in airand flue gas ducting in boilers",BHEL Nationalconference on CFD Application,Hyderabad, 17-18Nov,2006.*
10. *Yogesh Jaluria "Design and Optimization of ThermalSystem", Rutgers,The state University of New Jersey.*
11. *STAR-CCM+ software User Guide.CFD Analysis Of Economizer In A Tengential Fired BoilerInternational Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-4, 201255*
12. *Hryb,D,, Cardozo, M., Ferro, S., and Goldschmit M., 2009"Particle transport in turbulent flow using both lagrangianand Eulerian formulations",Int. Commun.Heat MassTransfer.*
13. *N. hedge, I. Han, T.W. Lee, R.P.Roy. (2007),"Flow andheat transfer in Heat Recovery Steam Generator". ASMEVol.129*
14. *Samuel, S., Iqbal, M., & Utama, I. K. A. P. (2015). An investigation into the resistance components of converting a traditional monohull fishing vessel into catamaran form. International Journal of Technology, 6(3), 432-441.*
15. *P.J. Stopford," Recent applications of CFD modeling in thepower generation and combustion industrie", Appl.math.Modell 26(2002)*
16. *Al-KhalidyNeihad, "Design optimization of industrial ductsusing CFD", CSIRO, 10-12 Dec, 2003, Australia.*
17. *M.EI Sayed et al (2005)-"Shape optimization withcomputational fluid dynamics",Advances in Engg. Software.*